PHASE I SBIR: INTEGRATION OF COMMON MODULE SIGNATURE CODE FEATURES

Allan R. Curran
Alan C. Koivunen
Keith R. Johnson

ThermoAnalytics, Inc.
94X Airport Rd
Calumet, MI 49913

Kurt Annen
Mark R. Anderson

Aerodyne Research, Inc.
45 Manning Road
Billerica, MA 01821

July 31, 1997

Final Report
Performance Period: November 1996 – May 1997
Contract: DAAEO7-97-C-X022
COTR: Ms. Teresa Gonda / TACOM
Security Classification: Unclassified
Approved for public release; distribution unlimited.

U.S. Army Tank-Automotive and Armaments Command
ATTN: AMSTA-TR-R/263
Warren, MI 48397-5000

REPORT DOCUMENTATION PAGE

Form Approved
OMB No. 0704-0188

1. AGENCY USE ONLY *(Leave blank)*	2. REPORT DATE	3. REPORT TYPE AND DATES COVERED
	31Jul97	Final: 04Nov96 - 04May97

4. TITLE AND SUBTITLE
Integration of Common Module Signature Code Features

5. FUNDING NUMBERS
C DAAE07-97-C-X022

6. AUTHORS
Allan R. Curran, Alan C. Koivunen, Keith R. Johnson, Kurt Annen, Mark R. Anderson

7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES)
ThermoAnalytics, Inc.
94X Airport Road
P.O. Box 66
Calumet, MI 49913-0066

8. PERFORMING ORGANIZATION REPORT NUMBER

9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES)
TACOM
ATTN: AMSTA-TR-R/263
Warren, MI 48397-5000
Contract Manager: Ms. Teresa Gonda

10. SPONSORING/MONITORING AGENCY REPORT NUMBER

11. SUPPLEMENTARY NOTES

12a. DISTRIBUTION/AVAILABILITY STATEMENT
Approved for public release: distribution unlimited

12b. DISTRIBUTION CODE

13. ABSTRACT *(Maximum 200 words)*

Report developed under SBIR Contract. This research program studied the feasibility of integrating common modules among leading signature codes. The codes under investigation included PRISM/MuSES, SPIRITS, and GTSIMS. The features that were candidates for integration included plume/flowfield codes, radiation view factor modules, and engagement models. A plan was developed for implementing ground vehicle and helicopter exhaust plume models into the signature codes. Significant progress was achieved in developing the architecture and code selection for ground vehicle plume modeling. Prototype software was produced to demonstrate the calculation and visualization of an exhaust plume on a ground vehicle within the MuSES code. The progress that was made in Phase I provides the basis for a signature design tool using the MuSES prototype. A comprehensive plan for Phase II focuses on further development of the universal design tool for virtual prototyping of concept vehicles.

14. SUBJECT TERMS
SBIR Report, Plume Code, Infrared Signature, PRISM, Ground Vehicles, Helicopters, Thermal Model

15. NUMBER OF PAGES
26

16. PRICE CODE

17. SECURITY CLASSIFICATION OF REPORT	18. SECURITY CLASSIFICATION OF THIS PAGE	19. SECURITY CLASSIFICATION OF ABSTRACT	20. LIMITATION OF ABSTRACT
Unclassified	Unclassified	Unclassified	SAR

NSN 7540-01-280-5500
(Rev 2-89)
18

Computer Generated

STANDARD FORM 298
Prescribed by ANSI Std 239-298-102

TABLE OF CONTENTS

Preface

The work was performed by ThermoAnalytics, Inc. under SBIR contract DAAE07-97-C-X022 for the U.S. Tank-automotive and Armaments Command (TACOM). The principal investigator was Dr. Allen R. Curran and the technical lead was Alan C. Koivunen. The subcontractor was Aerodyne Research, Inc. with Dr. Kurt Annen and Mark R. Anderson providing the technical support.

The contract was under the technical direction of Ms. Teresa G. Gonda of TACOM.

1.0 Introduction

The Phase I efforts concentrated on development of a plan for suitable exhaust plume models for ground vehicles and helicopters. Additional objectives included integration issues for PRISM/MuSES, SPIRITS, and GTSIMS. The project tasks as defined in the proposal included (1) IBS (Image Based Systems) Code Development, (2) IBS Code Interface Plan for SPIRITS, (3) Flow/Plume Code Interface Plan for PRISM/MuSES and SPIRITS, and (4) PRISM/MuSES and SPIRITS Code Interface Plan for use in GTSIMS.

The Task 1 code development goals are repeated here exactly as specified in the original proposal.

a. Initial implementation of the new IBS into PRISM/MuSES (this is expected to be near completion by the time of the award under current PRISM/MuSES development activities).
b. Add additional features that are in the Fortran IBS but not currently in the ANSI C version (this activity also is expected to be near completion by the time of award). The features that are required include incorporation of reciprocity, apparent solar/sensor areas, and subdivisions of emitters.
c. Develop new features including transparency factor, reverse side elements, utilization of 8 bit visuals when 24 bit graphics are unavailable, patching technology for large models, and extension of the available element primitives.

The predominant features from this list were implemented through other ThermoAnalytics projects and the remaining items will be proposed for Phase II.

The Task 2 IBS Code Interface Plan was determined to be more useful for the Aerodyne code GSL rather than SPIRITS. The implementation is postponed for Phase II activities. Task 4 activities which included a GTSIMS interface with SPRITS and MuSES has also been postponed for Phase II. Since the philosophy of MuSES is technical collaboration towards a universal design tool, both Task 2 and 4 can efficiently be achieved within the MuSES architecture. The plan is summarized in Section 4 and will be detailed in the Phase II proposal.

The Task 3 Flow/Plume Interface Plan is outlined in Sections 2 and 3 of this report. By concentrating our Phase I efforts predominantly on these task activities, the team was able to make significant progress towards the flow/plume model planning and implementation.

Two classes of plumes have been under investigation for flow modeling options, those of ground vehicles and those of helicopters. Variation in the flowfield physics and plume trajectories between the two suggest that two separate approaches be taken. The primary concerns in deciding which codes should be used are the accuracy of the results and computational efficiency. Section 2 describes the Phase I efforts for ground vehicle plume modeling and Section 3 provides the recommendations for helicopter plume model development.

2.0 Ground Vehicle Plume Model

The ground vehicle plumes are typically low temperature and very low inertia exhaust gases which generally exit from rectangular exhaust ports. Because the plume exhaust represents a smaller portion of the overall infrared signature in ground vehicles as compared to air vehicles, it

is important that the plume calculations do not take an excessively largely portion of the computational burden.

2.1 Plume Flowfield and Radiation Codes

Aerodyne Research Inc. has recommended the use of RNPF (Rectangular Nozzle Plume Flowfield) and RNPR/3D (Rectangular Nozzle Plume Radiation) to model ground vehicle exhaust plume flow and radiation, respectively. The RNPF and RNPR codes were developed in the mid-1980's and were originally used for analysis of new rectangular nozzle designs for high performance aircraft. Section 2.4 describes RNPF and Section 2.7 describes RNPR/3D.

2.2 MuSES Plume Model Plan

The plume model addition to MuSES presumes that all information normally needed to drive the PRISM signature model is available. In addition, information concerning either (a) exhaust gas properties at the exhaust system exit plane or (b) exhaust gas properties at the entrance to the exhaust system and the geometry for the exhaust system, will be needed to provide boundary conditions for the plume flowfield solver.

In case (a) above, RNPF can be run directly. In case (b), exit plane flow characteristics must be determined for input to RNPF. This may be done either with a MuSES run, utilizing a 1-D duct flow modeling capability to be added to the code or, avoiding a full signature model run, running a stand-alone 1-D duct flow modeling program. RNPF makes plume flowfield pressure, velocity, temperature, and composition predictions which will then be used in subsequent calculations of plume induced hardbody heating and plume IR signature. RNPF generates an "rnpfbox" file with the flowfield information.

After RNPF is run, the predicted plume flowfield geometry will then be combined with the vehicle geometry to determine regions of impingement of the plume on the vehicle hardbody and on the background. A new PRISM/MuSES vehicle geometry file will be generated, including a faceted description of the plume envelope. Also saved to files will be the rotation/translation transformation taking the RNPF plume flowfield into the vehicle coordinate system, allowing attachment of the plume to the vehicle exhaust exit nozzle, and the mapping of regions of plume impingement on the vehicle hardbody.

The signature model in MuSES will next be run, using RNPF output and the impingement region/plume flowfield mapping to control calculations of plume induced convective heating of the hardbody. The radiance output will be used in viewing the signature within MuSES.

RNPR/3D will be used to calculate plume radiances. It requires RNPF output and a specification of the lines-of-sight through the plume for which radiances are to be calculated. Since we will wish to view the hardbody and plume from any direction, radiances will be needed for a distribution of lines of sight over all angles (azimuth and elevation) and all points of intersection with the plume. Since RNPF assumes quarter symmetry of the plume, we need only examine one quadrant of the plume. The set of lines of sight will be determined using the plume envelope mentioned above and then written to a file, for use in driving RNPR/3D.

RNPR/3D is then run for the specified lines of sight and the radiance output saved for use in viewing the plume. Finally, the complete predicted signature may be viewed within MuSES. The

modified geometry file (including the faceted plume envelope), the RNPR/3D radiance file, the flowfield geometrical transformation file, and the MuSES hardbody radiance output file will be needed. Figure 1 provides the process flow of all of these modules as well as the interconnections of the input/output files.

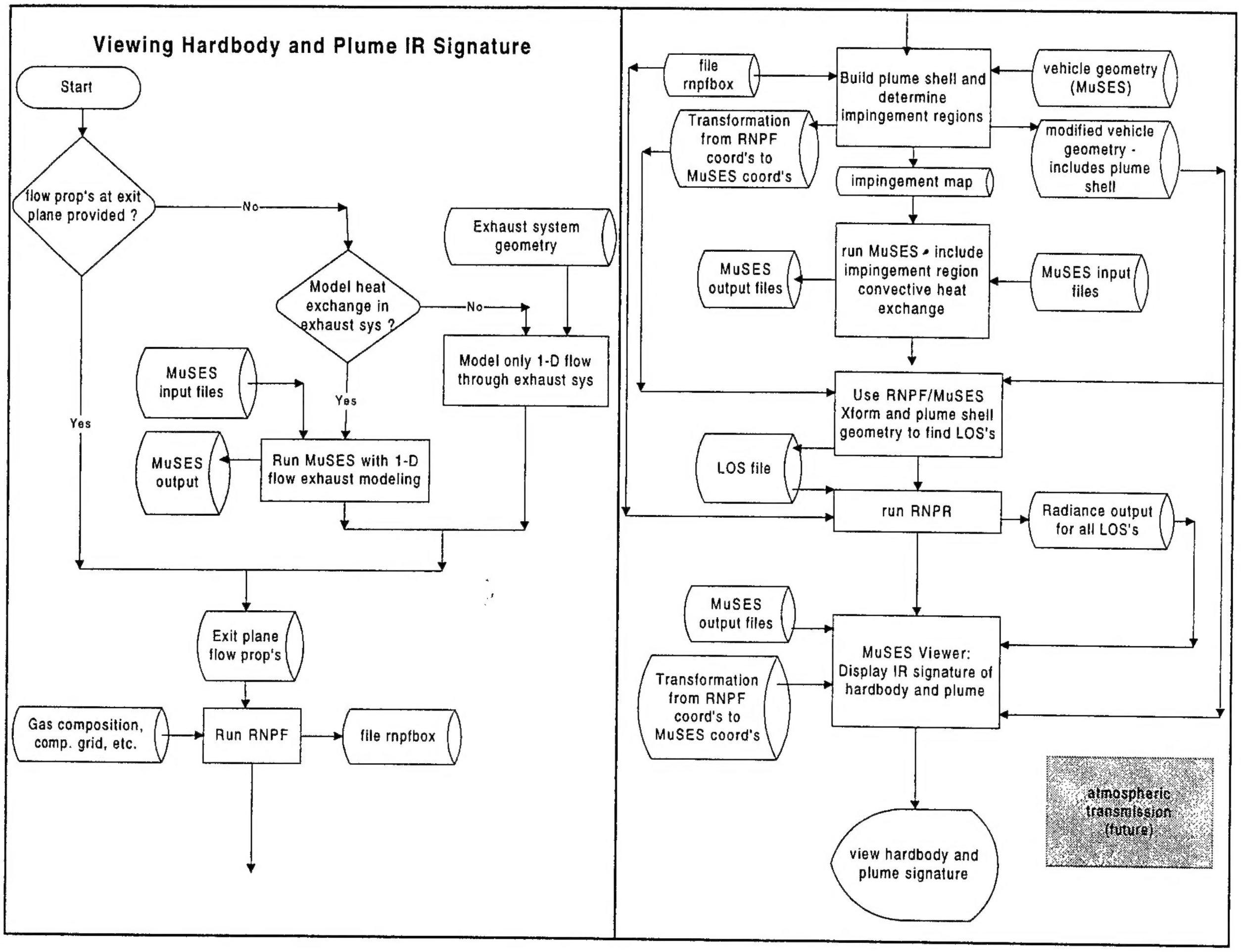

Figure 1. Program Flow of IR Plume Computations.

2.3 Exhaust System Modeling

If exhaust exit plane gas properties are not provided by the vehicle manufacturer, they must be predicted using exhaust system geometry information and gas properties (pressure, temperature, and composition) at the exhaust system entrance. A one-dimensional model using relations between pressure drops along duct sections and mass flow through the duct, adapted from ASHRAE literature, will be used to predict exhaust gas pressure and velocity at the exit plane. The flow model may be integrated within the thermal model of MuSES, which would also allow modeling heat exchange between the exhaust gas and the exhaust system surfaces and provide an estimate of the gas temperature change from the engine to the exit plane. This scheme would require running the signature model twice: (1) first to determine exhaust conditions at the exit plane and (2) again to model plume impingement heat exchange. A shorter procedure would require writing a stand-alone code to model gas flow through the exhaust system - but would neglect the exhaust gas/exhaust system heat exchange modeled in the signature model. If, however, the exhaust gas temperature is known at the exit plane, the 1-D flow stand-alone code

could be run to find exit plane pressure and velocity and the known temperature entered separately.

2.3.1 1-D Exhaust System Flow Code

Calculations of steady-state air flow through the exhaust system will be carried out using an implementation of ASHRAE's formulas for friction losses in gas flow through ducts.

Given exhaust system duct geometry - length, cross-sectional shape, and connectivity information, gas flow circuits through the network will first be identified. Each length of duct will be divided into sublengths or 'cells' for computational purposes. While exhaust gases are flowing through the network, pressure will drop across each cell as a result of the friction between the moving gas and the inner surface of the duct. These pressure changes are dependent on the shape and diameter of the duct cross-section, the length of the duct, the rate of gas flow through the duct, and the roughness of the inner surface of the duct. Duct diameter discontinuity and change in direction will also produce pressure drops and will be included in the calculations.

With the duct network geometry defined and the duct sections divided into cells, we make an initial guess at the set of flow rates in the ducts. With this seed value we then sum the pressure gains and drops across all flow cells in all flow circuits. The net change in pressure around any flow loop must be zero - so we embed this summing procedure inside of a root-finder and adjust the set of flow rates until all flow circuit pressure sums go to zero. The resulting set of flow rates will then be consistent with the exhaust system network.

2.3.2 Heat Transfer Within Duct Networks

Convective heat transfer between gases flowing inside the exhaust system and the system inner surfaces, heat conduction through the exhaust system walls, and convective and radiative heat exchange between the system outer surfaces and the environment will also be modeled. A single convection coefficient will be calculated for heat exchange between gas flowing inside the exhaust duct and the duct inner surface. The gas temperature inside of each duct cell will be estimated using this convection coefficient - tracking the change in gas temperature along the length of the duct. A one-dimensional heat conduction equation will be solved for heat transfer through the duct wall. The results of the flow and heat exchange modeling are estimates of the exhaust gas temperature, pressure and velocity at the exhaust system exit plane, which may then be used as input to RNPF.

2.4 RNPF (Rectangular Nozzle Plume Flowfield) Code

RNPF is the code to be used to predict plume flowfield characteristics. RNPF assumes a relatively simple co-axial flowfield where the ambient flow moves in the same direction as the plume exhaust. Such an assumption is necessary in order to preserve the computationally efficient quarter plane symmetry approach. A jet where the ambient flow has a perpendicular component of velocity will differ from a co-flowing jet in two key ways. First, the trajectory is no longer a straight line but the jet bends in the direction of the ambient flow. Second, the gross mixing rate tends to increase, leaving a corresponding decrease in the IR signature. Figure 2 shows a plot of centerline temperature of a low speed co-flowing jet compared to one with a 20

degree cross flow. The plot shows a definite enhancement of mixing in the 20 degree cross flowing jet as evident from the faster drop in peak temperature.

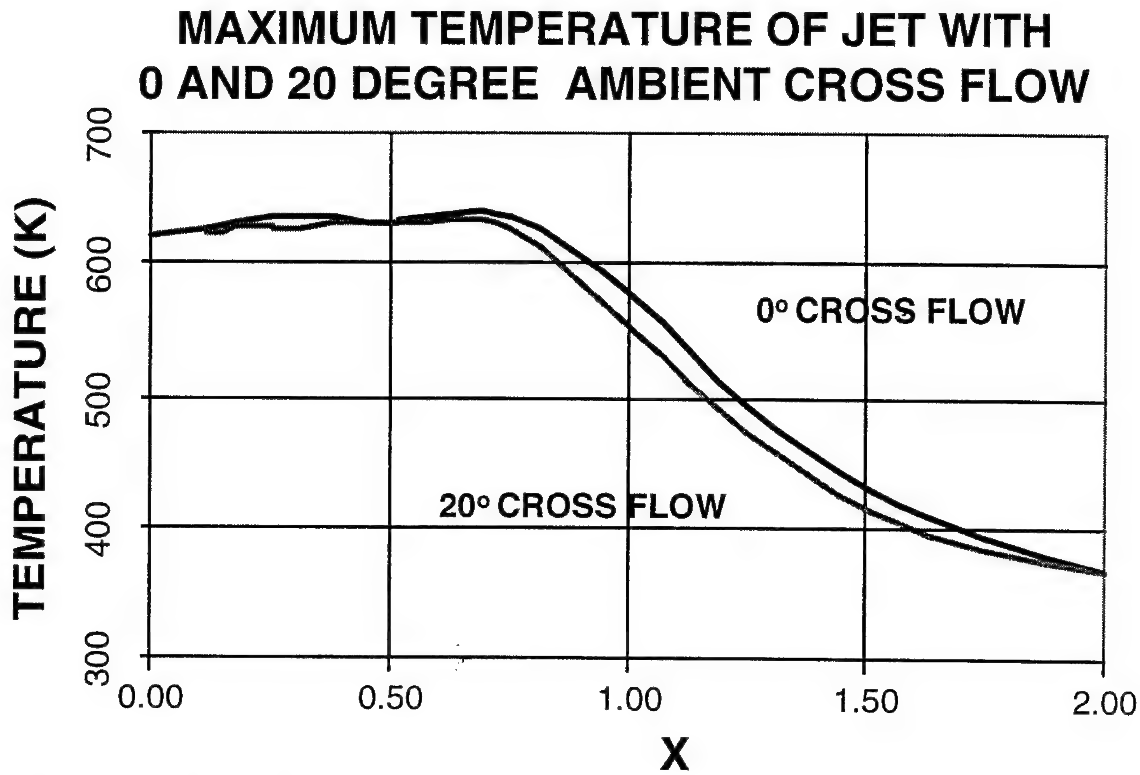

Figure 2. Plots of Maximum Temperature in the Cross Plane of a Co-linear and 20 degree Cross Flow Jet.

In terms of the infrared signature, the trajectory of the exhaust is generally not as critical as the mixing rate. Shifts in plume direction may bias certain lines of sight higher or lower than others but the total integrated signature is generally not strongly effected. Post-processing of the solution can be used to bend or shift the plume geometry if necessary before the infrared lines of sight are calculated. Because the IR signature is critically dependent on temperature, proper calculation of the mixing rate is extremely important. The problem is based on maintaining the efficient quarter plane symmetry while still accounting for changes in the flowfield resulting from three dimensional effects.

A technique that has been used very successfully in the past has been to maintain the same simplified computational approach but to alter the effective turbulent energy of the flowfield[1]. The process begins with a series of more advanced calculations (fully incorporating the three dimensional effects) run across a range of cross flow angles. The results are then tabulated to determine the degree of mixing enhancement, usually in terms of jet centerline temperature, as a function of axial distance. The turbulent energy input parameters for the simplified analysis are then varied until it produces similar results to that of the more complex model. This empirical/analytical approach has been found to work extremely well for plumes in moderate

[1] "Three Dimensional Flowfield and Radiation Modeling for the C-130H Aircraft," E. Strang, K. Annen, IRIS Meeting, JANNAF Conference, 1993.

cross flows[1,2]. The empirical aspect of the approach limits the effectiveness to a narrow range of jet and ambient conditions though. Because of this, several inputs must be calculated which span across the operating range of the given vehicle. Figure 3 shows an example of this approach where a two dimensional model was modified for three dimensional effects for the Air Launched Cruise Missile plume.

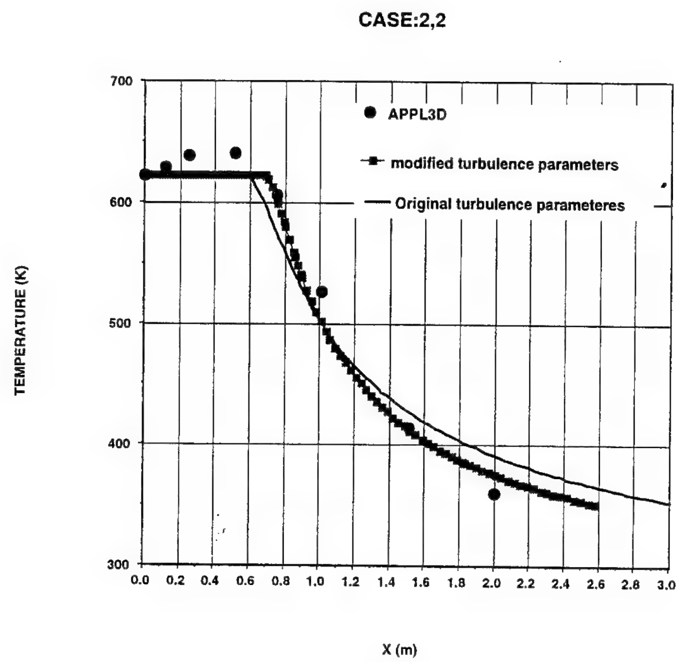

Figure 3. Plot of Two Dimensional Turbulent Enhanced SPF-II Plume Calculated to Match for Three Dimensional Effects.

The severity of the cross flow effect on a jet depends on several factors. The effects increase with the cross flow angle, ratio of ambient to initial jet momentum, and turbulence/unsteadiness in the ambient. Unsteady effects are particularly prevalent in low momentum jets such as ground vehicle exhausts. Examination of a car exhaust on a cold day clearly demonstrates the results from wind gusts and vortex shedding off of the car body. This process causes the very transient and billowing phenomenon commonly seen at low speed. Transient effects per se are generally too complex to model directly but can be simulated in enhancements to the turbulence model.

An example of an RNPF input file is shown in Table 1. The input parameters are straight forward and the inherent stability of the code lends a high degree of usability. This is especially true for users without Computational Fluid Dynamics (CFD) experience. Figure 4 shows a plot of CO_2 concentration at the center plane for a jet engine exhaust calculated with RNPF. Figure 5 shows several cross-sections of a jet which clearly demonstrates the gradual transition that the exhaust undergoes from rectangular to circular cross-sections.

[2] "First Principles Modeling of Turbine Engine Hot Parts and Nozzle Mixing Using the TRBEXT Module," K. D. Annen, M. R. Anderson, L. F. Kelly, JANNAF SPIRITS User Group Meeting, 1994.

Table 1. RNPF Input File

```
*************************************************************************************
2/1   nozzle test case
4              initial no. of grid points (nzx max.)
4              no. of species
-1             viscosity option -1 k-epsilon 1 model, -2 k-epsilon 2 model
64             no. of strips inside the shear layer
9              no. of boxes for radiation calculation
0.             initial value of x (ft)
60.            final value of x   (ft)
300.0          temperature to stop calculation at (tstop)
5.             print increment    (ft)
20.            x for new print interval (ft)
10.            new print interval (ft)
0.60           half height of nozzle -ft
1.20           half width of nozzle -ft
1.             divide box by variable to average properties
2.             boxl=variable*plume half width
.521           pressure in plume (atm)
650.0 230.0        temperature jet centerline, freestream (K)
287.42  80.05      velocity jet centerline, freestream (ft/sec)
O2
0.1350,  0.21      *** o2  jet , outside
H2O
0.0418,  0.0       *** h2o jet , outside
CO2
0.0400,  0.0       *** co2 jet , outside
N2
0.7814,  0.79      *** n2  jet , outside
*************************************************************************************
```

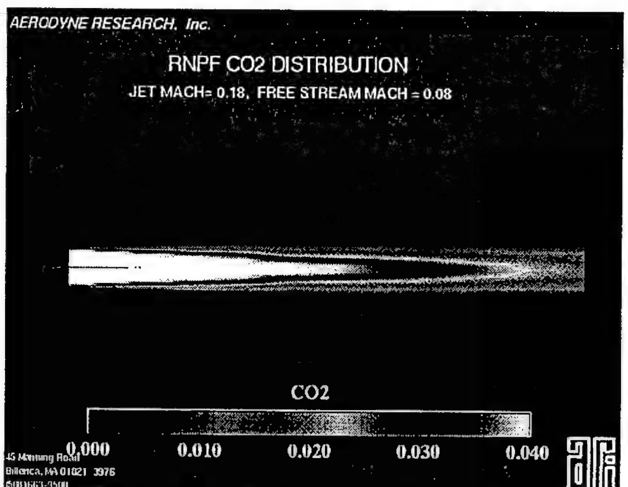

Figure 4. CO_2 Distribution of an Exhaust Jet Calculated by RNPF.

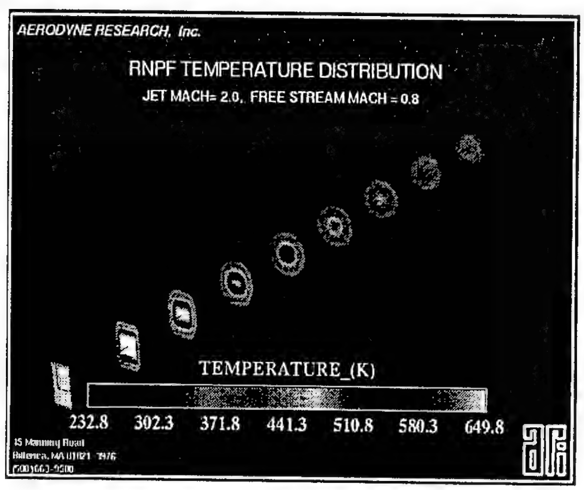

Figure 5. A Jet Transitioning from Rectangular to Circular Cross-Section Calculated by the RNPF Code.

2.4.1 Assumptions

There are two classes of assumptions built into the RNPF flow model. The first are those inherent in the computational procedure, and the second are the prescribed features of the flow,

which in RNPF's case lead to the prediction of the non-axisymmetric features of the flow. The assumptions of the first type are:

a. The flow can be described by the constant pressure mixing of a moving ambient air stream with the exhaust plume.
b. The mixing between the plume and the air can be described by the Reynolds averaged equations of motion using an eddy viscosity derived from an equation for turbulent kinetic energy.
c. The turbulent Prandtl and Lewis numbers are unity.
d. The mixing is chemically frozen. This assumption, while not strictly necessary, is valid for turbine engines where there is negligible unburned fuel in the exhaust.
e. The flow is adiabatic.
f. There are no effects of the pressure field, boundary layers, or vortices from the vehicle included in the model.
g. The exhaust composition at the exit plane is uniform.

The basic procedure in converting the non-axisymmetric flowfield into an equivalent axisymmetric one requires a knowledge of the shape of the plume during its transition from a rectangular cross-section at the nozzle exit to a circular one in the far field. This involves three major assumptions:

a. The unmixed core of the plume is rectangular in cross-section. The mixing zone surrounding this rectangle has lines of constant properties parallel and equal in length to the sides of the core and connected at the corners by circular arcs. At the end of the core, the core rectangle has degenerated into a line of constant properties parallel to the long axis of the nozzle . The lines of constant properties in the mixing zone are parallel to this line and connected by semicircles at the ends. As the mixing progresses, the parallel lines of constant properties become shorter and eventually the plume becomes circular in cross-section.
b. The data of Sfeir[3] indicates that for a nozzle aspect ratio of 10, the width of the plume changes very little until the cross-section becomes circular, and that the growth occurs in the direction parallel to the short side of the nozzle. The plume from a square nozzle would be expected to grow equally in both directions. An empirical relationship is used to force this effect to occur in the model.
c. The mapping between the axisymmetric calculation and the plume from the rectangular nozzle is based on the assumption that the mass flow inside a given line of constant properties on any cross-section is unchanged in the transformation.

2.4.2 Program Input

Inputs specifying flow conditions at the exit plane will be supplied by the new MuSES exhaust system flow model and engine model. This information will include nozzle geometry, gas temperature, composition, pressure, and velocity at the exit plane. Specification of the computational grid will require the user to supply either plume length or plume 'cutoff' temperature, the fraction of the plume in the mixing layer, the number of horizontal and vertical

[3] Sfeir, A.A., "Investigation of Three-Dimensional Rectangular Jets", AIAA Journal, Vol. 17, Oct 1979,. Pp 1055 – 1060.

boxes at each point along the plume axis, and the number of computational points along the plume axis.

2.4.3 Program Output

The flowfield properties for use by MuSES and for radiation calculations in RNPR/3D are written by RNPF to file “rnpfbox”. The output contains the plume properties mapped to rectangular volumes (boxes). The number of horizontal boxes is the same as the number of vertical boxes, and both are constant through the length of the plume. At a given point along the plume axis (an “axial station”) all boxes are the same size and shape, although box sizes will change with axial position.

At each station one box is centered on the plume centerline. Since there are two planes of symmetry, the properties are given for one quadrant only. The file specifies the box dimensions and positions at each axial station, the mole fractions of CO_2 and H_2O in each box, and the gas temperature and pressure inside each box.

2.5 Post-RNPF, Pre-Signature Model Processing

Given the RNPF computed plume flowfield, we next determine the impingement of the plume on the ground and on portions of the vehicle. The impingement geometry and the plume temperature and velocity can then be used to compute convective heat exchange between the plume and the objects on which it impinges. Pre-processing of the plume and vehicle geometries will be carried out and the resulting plume/hardbody impingement mapping written to a file. When the signature model is run, the plume impingement mapping file and the RNPF flowfield file can then be used to control convective heat exchange calculations between the plume and the vehicle hardbody.

2.5.1 Plume-Background Impingement

The RNPF output file organizes the above mentioned ‘boxes’ so that the plume axis is increasing along the x-axis, with the plume origin at x=0. The y and z axes will be placed parallel to the long and short dimensions of the rectangular nozzle, respectively, forming a right-handed coordinate system. The center of the exhaust nozzle then corresponds to the origin of the plume, and the plume geometry can be translated and rotated to position on the hardbody. The ground plane, in the vehicle world coordinate system, is the z=0 plane, and intersection of the plume with the ground occurs at those plume boxes with corner coordinates having both positive and negative z values - those boxes intersecting the ground plane. The projection of those boxes on the ground plane then determines the location of plume-ground convective heat exchange, and the gas velocity and temperature within those boxes can be used for the convection calculation.

2.5.2 Plume-Hardbody Impingement

Determination of the region of plume impingement on the hardbody is not as straightforward as the ground impingement, since there is no simple constant coordinate position at which impingement occurs. One possible approach utilizes OpenGL graphics capabilities to recognize hardbody-plume intersections. In this approach, the plume geometry is obtained from the RNPF output file and then translated and rotated into position at the exhaust system exit plane. An

'observer' is graphically placed at the center of each plume box and looks outward toward each of the six sides of the box. Viewer clipping planes are placed so that the viewer can see only from the center to each of the sides, and not beyond the side. Hardbody polygons visible from inside the plume box then lie within that box and will be subject to convective heat exchange with the plume gas within the box.

In Figure 6 the exhaust nozzle is at the bottom of the figure. The dashed lines represent the plume 'boxes' used by RNPF. The shaded elliptical volume on the west side of the figure represents the part of the hardbody on which the impingement occurs. In plume box (3,3), the observer, looking west, sees no hardbody polygons, implying that there will be no convective heat exchange between the hardbody and the gas in box (3,3). In box (3,1), the observer, looking east, sees the backside of polygons forming the hardbody, so the gases in plume box (3,1) will exchange heat with the hardbody. The identity of the particular plume box (in this example, box (3,1)) enclosing the polygons will be saved so that when the thermal solver is run, convective heat exchange between these polygons and the gas in box (3,1) can be computed. In order to avoid considering hardbody polygons that are completely shielded from the plume, the *EXT.fac file (external polygons only) will be used to determine plume/hardbody impingement.

This approach is very approximate since the estimated plume flow-field will have been calculated with the hardbody absent and won't reflect the influence of the hardbody on the flow-field itself.

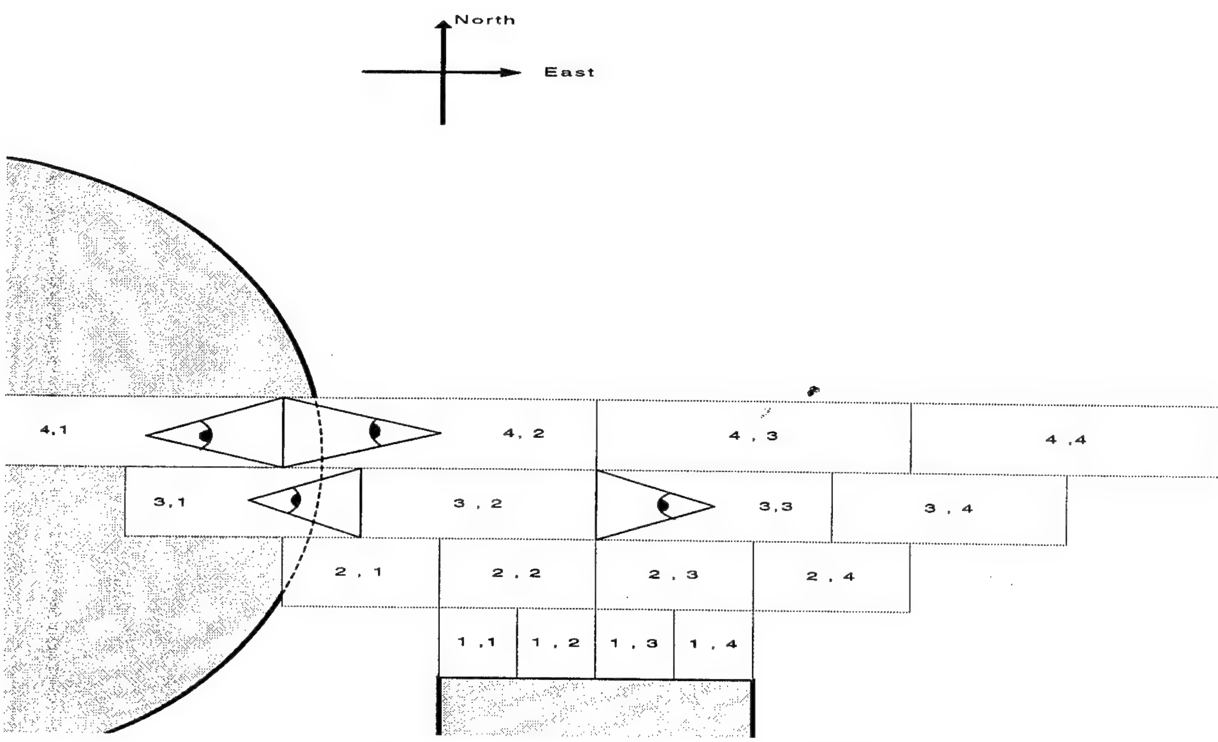

Figure 6. Plume Impingement on Hardbody.

2.6 MuSES Infrared Signature Processing

With plume geometry, temperature and velocity calculated, and the regions of plume impingement on the hardbody determined, MuSES can then be run. The MuSES output will reflect the effects of convective heat exchange between the plume and the regions of plume impingement on the hardbody.

2.7 RNPR/3D - Plume Radiation Calculations

MuSES provides the information necessary to display the hardbody IR signature and RNPR/3D will be used to find the plume's signature. In order to allow the user to quickly and easily view the hardbody and plume signature from any point of view, a means must be devised to quickly calculate the plume signature for any line of sight through the plume from the results of a limited number of RNPR/3D line of sight radiance calculations. A straightforward bilinear interpolation between RNPR/3D calculated lines of sight is suggested. In order to decide on the particular lines of sight to be used in the plume radiance calculation, we must first decide how the plume will eventually be displayed with the hardbody.

2.7.1 Plume Envelope

RNPF maps the plume flowfield into sets of rectangular volumes (boxes) with a value of temperature, pressure, CO_2 mole fraction and H_2O mole fraction assigned to each box. The boxes form an (n x n) array with all the boxes at a given axial station being the same size and shape. The center of box (1,1) is on the centerline of the plume and the boxes are numbered outward from this point. The size of the boxes increase with axial distance from the origin to accommodate the growth of the plume.

We propose to display the plume by building a faceted envelope around the plume and base the shading of the envelope polygons on RNPR/3D plume radiation calculation results. We plan to fit ellipses around the set of boxes at each axial station. Since the plume expands with increasing distance along the plume axis, the ellipses will enlarge as well. The ellipses will be fit so that their perimeters lie entirely outside the set of boxes at each axial station. Polygons would then be used to form a surface joining the set of ellipses - enveloping the plume.

In Figure 7 we see the set of boxes in one quadrant of the plume at a particular axial station, and the circumscribing ellipse. Note that as we go from LOS 2 to LOS 1 the radiance will be discontinuous as the line of sight changes from passing through the plume to missing the plume - although the radiance for LOS 2 may be very small. This discontinuity can be reduced in the display by using Gouraud shading (also see section 2.8) to smoothly vary the shading across the envelope polygon that spans the discontinuity. Since LOS 1 follows a path through the envelope that completely misses the plume, we will assign an opacity of zero to the polygons in that part of the envelope. Using Gouraud shading with opacity=0 makes the polygon transparent and allows the object behind the plume to 'shine through'. Note that LOS 3 passes through the same point on the plume envelope but also passes through the plume. In this case the envelope polygon would no longer have zero opacity and the polygon shade would be based on the RNPR/3D computed plume radiance.

It's not clear that a circumscribing ellipse is the best shape for the envelope to take. It is inefficient in that a large number of lines of sight do not pass through the plume and this 'null

information' would have to be saved for use in plume shading - to flag for polygon transparency. The ellipses could be built to lie completely inside of the boxes - inscribed within the plume. In this case, however, some lines of sight that *do* pass through the plume would not be represented. For example, LOS 4 of Figure 7 passes through the plume but not through the inscribed envelope. In this case the display would incorrectly indicate no plume radiance for this line of sight.

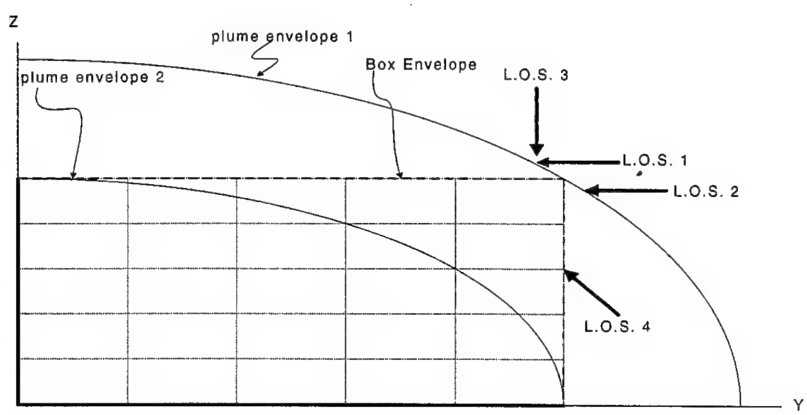

Figure 7. Cross-Section of One-Quarter of Plume 'Boxes' with Possible Elliptical Envelopes.

Another envelope shape candidate is depicted in Figure 8. This envelope simply follows the box surfaces except at the corners, where a portion of an ellipse is used to round off the box. This shape is more efficient in that many fewer lines of sight miss the plume than in the case of the circumscribed ellipse. Finally, a combination of the two may be used - a rectangular envelope cross-section near the exit plane and an elliptical or circular cross-section near the end of the plume.

We anticipate displaying the plume by assigning radiances to envelope polygon vertices by interpolating into an RNPR/3D generated radiance table. Generation of this table will require a set of lines-of-sight at which RNPR/3D is to compute radiances.

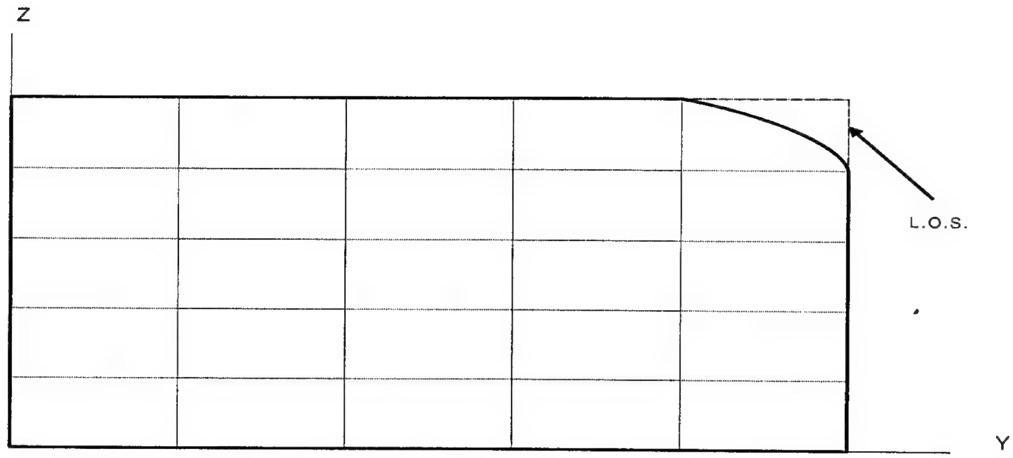

Figure 8. Cross-Section of One-Quarter of Plume 'Boxes' with Possible 'Rounded Box' Envelope.

2.7.2 Line of Sight Specification

Since the plume has quarter-symmetry, we need specify lines-of-sight through only one quadrant of the plume, for its entire length. The points of intersection of the lines-of-sight will be the vertices of the polygons forming the plume shell. At each of the points of intersection, radiances for multiple lines-of-sight will be calculated - over a range of 'elevation angles' (rotation about the plume axis) from 0 to 90 degrees, and a range of 'azimuth angles' (line of sight varying from directly in front of the plume to directly behind the plume) from 0 to 180 degrees. RNPR/3D currently allows the user to specify one observer position with respect to the plume and then computes radiances for lines of sight that are spaced on a regular grid in the image plane. The above requirement that the radiances be computed for lines of sight passing through the envelope vertices means that RNPR/3D will have to be modified to allow lines of sight that are irregularly spaced in the image plane. RNPR/3D will also be modified to allow multiple runs over the range of elevation and azimuth angles. These modifications will be done in collaboration with Aerodyne Research Inc. The resulting table of radiances will then be saved and interpolated into when 'coloring' the plume during display within MuSES.

2.8 *Plume and Hardbody Display*

The plume flowfield is described with the coordinate system of RNPF. RNPR/3D uses another coordinate system to describe the plume when computing line of sight paths through the plume. MuSES (PRISM) use a third coordinate system for thermal calculations and display. A coordinate transformation from RNPF to MuSES will be needed to place the plume envelope, developed in the RNPF coordinate system, into position in the MuSES coordinate system. Another transformation, from the MuSES coordinate system to the RNPR/3D coordinate system, will be needed to map line of sight vectors in MUSES back to line of sight angles in RNPR/3D. The point of reference for determining the transformations will be the center of the exhaust plume at the exit plane. The exhaust nozzle in the RNPF and RNPR/3D coordinate systems will be rotated and translated to attach the plume to the exhaust nozzle in the MuSES coordinate

system. These coordinate transformations will be saved to files for use in display of the plume/hardbody combination.

2.8.1 Display Of RNPR/3D Determined Radiances

With the hardbody and plume radiances available, the combination is ready to be displayed. The hardbody/plume combination geometry file may be placed in viewing position using the Muses Viewer (Figure 9). With the object positioned, the OpenGL function *gluUnproject* can be used to obtain the unit vector corresponding to the viewer's line of sight in the MuSES coordinate system. The vector may then be transformed to the RNPR/3D coordinate system and the viewer's azimuth and elevation angles computed. Each of the plume envelope vertices will have radiances tabulated for the possible range of elevation and azimuth angles. We will use the *gluUnproject* determined viewer angles in a bilinear interpolation to estimate radiances at each visible plume envelope vertex. With the vertex radiances determined, the plume envelope polygons can then be shaded using OpenGL's Gouraud shading capabilities.

2.8.2 Use of RNPR/3D Determined Transmittances

The representation of radiation originating at objects behind the plume and passing through it to an observer is somewhat more difficult. RNPR/3D computes the total transmittance spectrum (across the plume) for each line of sight. These transmittances could be saved along with the radiances for each line of sight and then used to calculate the background radiation passing through the plume.

In order to use the transmittance information to estimate radiation passing through the plume, the line of sight would have to be ray-traced to determine the object it intersects behind the plume and the radiation spectrum of the radiating object would have to be known. The radiation spectrum could be multiplied by the 'transmittance spectrum' and the product integrated over the band of interest. The resulting radiance estimate could then be added to the radiance of the plume itself (determined by RNPR/3D) to produce an estimate of the total radiance for the line of sight.

Difficulties arise in a couple of ways: (1) The spectrum of the radiating object would have to be known - either with the use of a table of emissivities by wavenumber or by use of a fitted curve. (2) The transmittance spectrum would have to be saved at the desired spectral resolution for each set of azimuth and elevation angles at each envelope vertex - resulting in a very large file of transmittances for each set of operating conditions.

The code could be written to allow the user to include or neglect the radiation passing through the plume - only in the former case would the additional transmittance and emissivity by wavenumber information be needed.

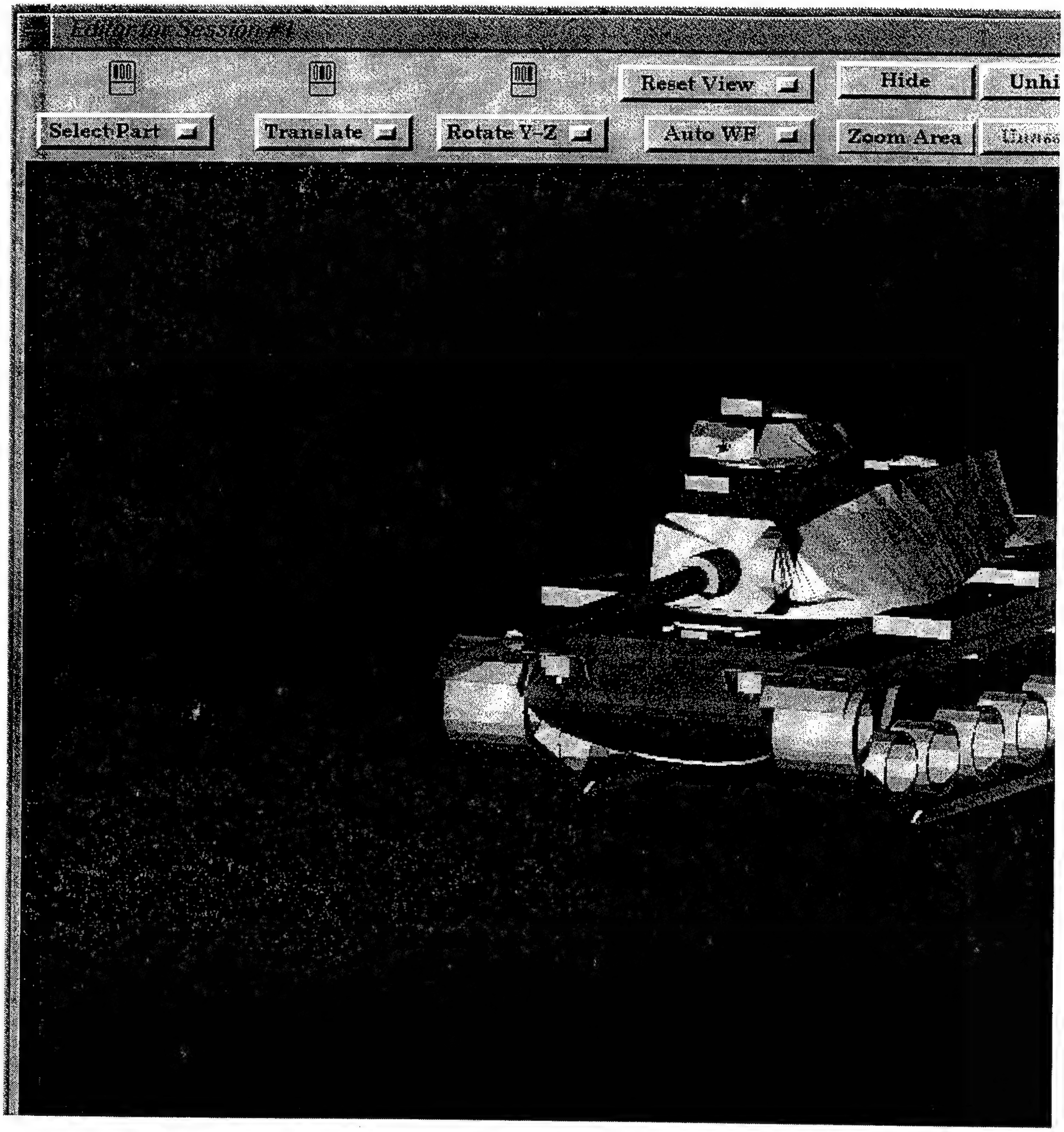

Figure 9. Example of Ground Vehicle Plume Displayed in MuSES.

3.0 Helicopter Plume Model

Helicopter plumes represent a more problematic calculation then ground vehicle plumes due to much stronger cross winds typically present. The cross velocity correction approach used in the ground vehicle analysis would be insufficient here since the cross winds the jets experience are often very severe. The higher portion of the IR signature resulting from the plume in helicopters also warrants a more rigorous approach. Several candidate solutions were investigated.

3.1 APPL/3D Code

The APPL/3D code developed at AMTEC in 1989 has the capability to calculate three dimensional deflecting jets in time efficient manner[4]. APPL/3D solves the 3D compressible parabolic/hyperbolic Navier-Stokes equations using a two-equation turbulence model and transport equations for chemical species. An implicit space marching procedure is used to advance the solution in the direction of the plume centerline trajectory using an alternating direction implicit algorithm followed by a pressure-continuity correction relation for stability in subsonic flows. APPL/3D can use an internal mesh generator which automatically shifts in the direction of the jet trajectory and the grid can be either radial or rectangular in cross section. A user defined grid can also be used in cases of severe jet bending or where problems of code instabilities are found. Multiple tracking of chemical species is provided in the code with the option for an equilibrium chemistry calculation based on a subset of the Gordon and McBride equilibrium chemistry model. Figure 10 shows a center plane temperature distribution of an exhaust plume for the Apache helicopter at hover generated with the APPL/3D code.

Three cost scenarios have been quoted by AMTEC for use of the code and are as follows:

1. Single-user floating license: $7,000 per copy;
2. Site license: $35,000 for up to 10 users on a single network;
3. General Distribution: $150,000 for distribution to any government agency. This option would include source code.

The costs for use of the code are high. Other more general problems in generating reliable and trouble free results with this code are also serious issues, in particular, for users not experienced in CFD. For these reasons, it was decided not to use the APPL/3D code for this effort.

[4] APPL/3D User's Manual, Amtec Engineering Inc., AEI-T90130.01, 1990.

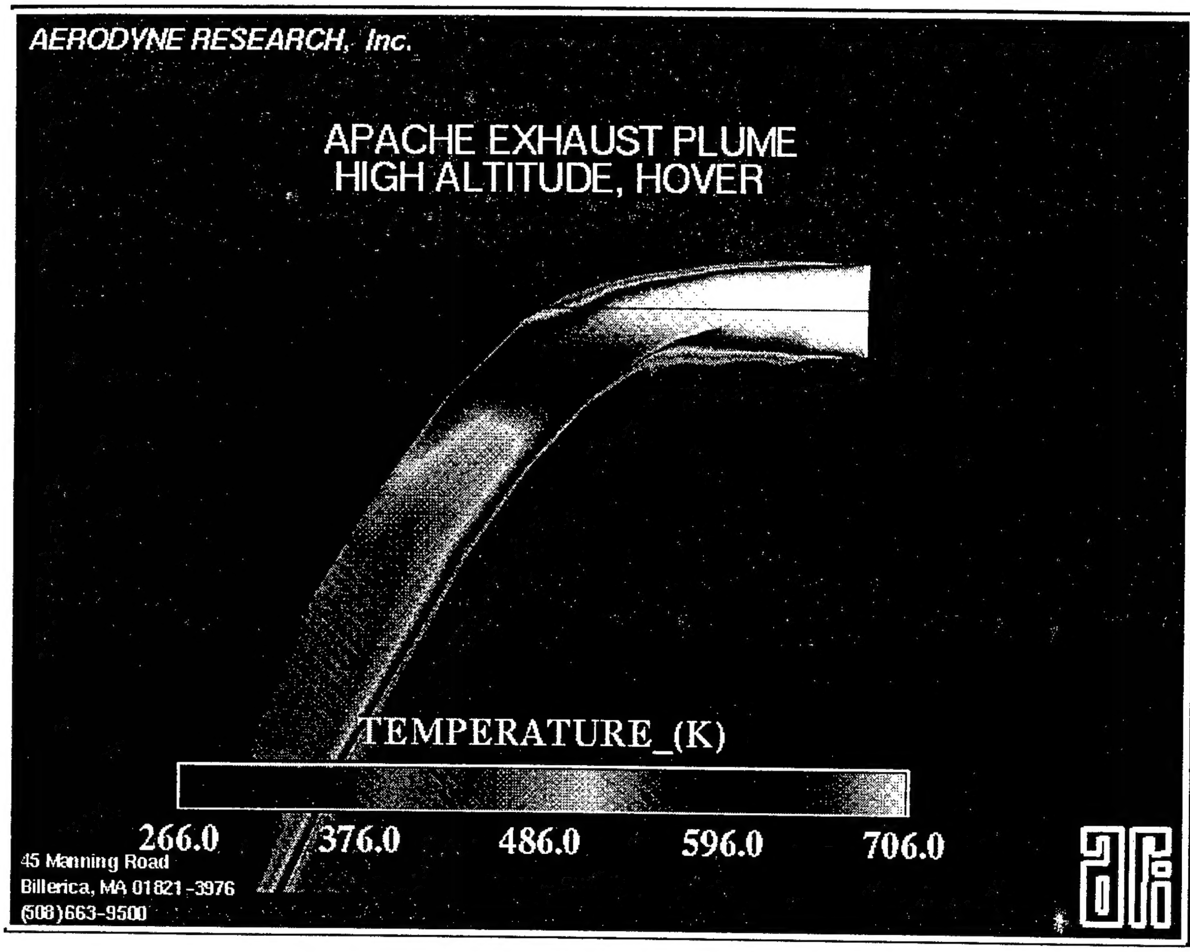

Figure 10. Plot of a Helicopter Plume Flow Deflected by the Rotor Downwash as Calculated by the APPL/3D Code.

3.2 STUFF CFD Code

The most rigorous and most expensive (developmentally) approach considered involves modifying an existing CFD code called STUFF to calculate helicopter plumes[5]. The STUFF code is a Parabolized Navier-Stokes (PNS) version of a code derived from the Full Navier-Stokes (FNS) code called TUFF. Both codes were developed at NASA Ames. The codes calculate the Reynolds averaged Navier-Stokes equations using a TVD (Total Variational Diminishing) algorithm developed for efficient shock capturing in high speed flows. Zero, one and two equation turbulence model options are available. The scheme is third order accurate in the special domain and uses approximate factorization for efficient computation for three dimensional flows. Unlike most PNS codes, STUFF has the ability to repeatedly sweep over a flowfield and thus can incorporate limited downstream effects on the flow. Tracking of multiple species with finite rate chemistry models is included in the code as well as an option for a equilibrium air equation of state for real gas effects.

[5] "A Set of Strongly Coupled Upwind Algorithms for Computing Flows in Chemical Nonequilibrium," G. A. Molvik, C. L. Merkle, AIAA Paper 89-0199, Jan. 1989.

TUFF and STUFF were originally developed for higher speed predominantly supersonic flows. To make the code fully functional for lower speed helicopter plume analysis, modifications to the code for constant pressure mixing would be required for it to run with stability. These changes would involve modifications to the pressure terms in the equations which would be invoked after each computational marching step. Also, the capability for automatic grid generation and jet tracking would need to be included to make the code fully functional for calculating deflecting plumes. Several discussions were held with the code author who assured us that the code was amiable to such modification and that the time estimates for altering it were reasonable. Since the codes are government-owned, no licensing or additional fees would be required. Although, most of the code would not require modification, the upgrades would still be substantial. A budget of approximately $150K was estimated for the effort. Part of this budget was projected to go the code author who stated he was available to consult on the effort.

3.3 Pre-computed Solutions

The third option, and the one determined to be best for the effort, involves using a series of pre-computed solutions as a database of plume flowfields. The helicopter exhaust plume is characterized by an ever present downwash from the rotor blades. Engine conditions are less variable than that of fixed wing aircraft since the engine must always work to keep the vehicle suspended. For these reasons, a helicopter's plume tends not to vary too significantly across the operating range. With this in mind, it would be possible to use a relatively small number of pre-computed solutions in a database to characterize the plume variation.

The specific conditions that the end user requires (in terms of operating conditions), would simply be interpolated from the pre-computed solutions. The draw back of this approach is that plume flows do, in fact, vary quite significantly for different helicopter designs. Therefore, a new series of solutions will need to be generated for each vehicle.

One advantage that the pre-computed solutions offers is computational efficiency, since the new solution is interpolated and not computed from scratch each time, only a handful of pre-computed solutions are required for each vehicle. More accurate and time consuming solutions can be generated than would otherwise be possible. It is proposed that the precomputed solutions use a FNS approach for maximal accuracy.

FNS calculations will allow the capture of downwind effects on the plume which can be non-trivial at low speed, particularly for highly deflected flows. Effects such as plume impingement and surface heating can also be calculated with FNS whereas PNS calculations can not. The NPARC code developed at AEDC would be used for these calculations. NPARC has been used successfully in the past to calculate complex helicopter plume flowfields[6]. Figure 11 shows a extreme example of a flow calculated across the OH-58D helicopter where plume impingement is occurring.

[6] "Computational Fluid Dynamic Modeling of an Exhaust Plume Impingement on an OH-58D Helicopter," M.R. Anderson, L. Kelly, M.R. Wohlers, K.D. Annen, JANNAF Conference, Exhaust Plume Technology, Huntsville, AL, 1995.

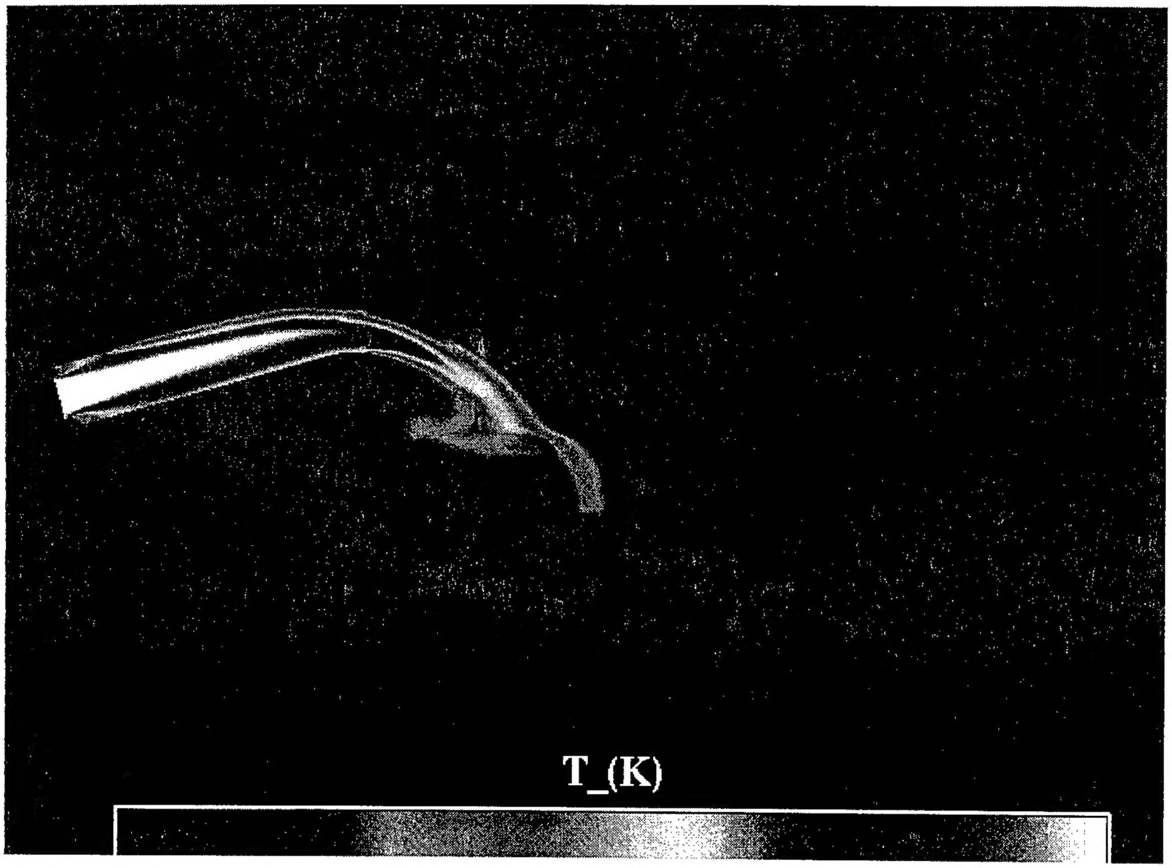

Figure 11. Temperature Distribution on the Center Plane and Body of a Complex Helicopter Plume Impingement as Calculated by the NPARC Code.

In CFD modeling, the bulk of the cost is often in the preprocessing stage of grid generation. These costs will be substantially reduced by the fact that all the solutions for a given helicopter can be run on the same computational grid. Most of these grids will also be fairly simple since plume impingement is generally not an issue.

A series of 8 CFD solutions should cover the entire flight range of a helicopter. These solutions will cover all of the variation in vehicle altitude, flight velocity, and engine power setting.

The completed solutions will be interpolated directly onto the RNPR files. Here too, there is an advantage to precomputed solutions. Finer computational grids in the precomputed solutions will lead to a more accurate interpolation from the curve linear to the rectangular RNPR grid. The pre-mapped IR files will be used as the database and the actual CFD solution files will be not be required. This approach will eliminate the interpolation step for the end user which can be a difficult calculation where substantial bending and drift of the plume spreads the calculation region over a large volume.

4.0 Plans for Phase II

The activities for Phase II will be detailed in the Proposal. A summary list of activities that were derived from the Phase I study are outlined below.

1. Finalize the implementation of the Ground Vehicle Plume Codes (and visualization) into MuSES.
2. Implement an atmospheric transmission (and path radiance) module (MODTRAN) into MuSES to support the total hardbody/plume IR signature computations.
3. Integrate a 1-D Flow Solver into MuSES so that both exit plane conditions can be determined for plume modeling and exhaust duct heating can be included in the thermal model.
4. Develop an interpolation scheme for extracting pre-computed helicopter flowfield solutions from a database and applying them directly on the rectangular RNPR grid.
5. Finalize improvements to the IBS code including transparency, patch technology, and additional geometric primitives.
6. Integrate the new IBS code into MuSES.
7. Provide an additional capability to build and export SPIRITS and GTSIMS (GTSIG) models from within MuSES so that they can use the new IBS module and other improvements.